AN2797
Application note

PCB layout guidelines for SPEAr600

Introduction

SPEAr600 is a 23 x 23 mm PBGA420 device with 1 mm ball pitch. It is a member of the SPEAr® family of 32-bit embedded MPUs. The device features dual ARM926 cores running at up to 333 MHz, an external DDR2 memory interface and a full set of powerful on-chip peripherals.

This application note provides guidelines for successfully designing the PCB layout for SPEAr600. It covers following main topics:

- Power integrity
- DDR memory interface
- USB signals
- GPIO/CLD signals
- External clock

Placement guidelines for the decoupling capacitors are provided in *Appendix A*.

Contents

1 Power integrity 4

1.1 PCB stack up 4

1.2 Via padstack 4

1.3 Part orientation and placement 4

1.4 Ground and power supply connections 5

2 DDR memory interface 6

2.1 DRAM power decoupling 6

2.2 Data signal routing 7

2.3 Trace length matching 8

2.4 Trace length matching, dual DRAM configuration 10

2.5 Return path integrity 11

2.6 Clock routing 12

2.7 Vref routing 12

2.8 Observability 13

3 USB interface 14

3.1 USB routing 14

3.2 USB decoupling and reference resistor 15

3.3 USB Device Vbus 16

4 GPIO and CLD interface considerations 17

5 Using an external clock 18

5.0.1 External clock signal requirements 18

5.0.2 Voltage translation of external clock 18

5.0.3 TDR test traces 19

5.0.4 Layer order check 19

Appendix A Low-inductance decoupling capacitor layout 20

A.1 0402 compact land pattern 20

A.2 Low inductance layout for decoupling capacitors 21

Revision history 22

1 Power integrity

One of the most important requirements of a reliable high-speed memory interface, and most commonly underestimated, is a low impedance, wide bandwidth power supply at the power and ground balls of the devices. Achieving the necessary performance requires the minimizing of all parasitic inductances found in power delivery and grounding connections, exploiting various techniques to provide low impedance paths, and attention to controlling plane resonances.

A solid unbroken ground plane located close to the high-speed devices in the PCB layer stack is critical. The ground plane must not have large gaps anywhere in the area of the interface. Be especially aware of overlapping anti-pads that can create an extended gap in the internal plane layers.

A power plane closely spaced to the ground plane greatly aids the high-frequency decoupling by providing a low inductance path between a capacitor and the device's power balls.

All high-frequency decoupling capacitors should utilize a low-inductance layout.

1.1 PCB stack up

Stack up should include a closely spaced power/ground plane pair. A minimum of 6 layers is necessary.

Recommendation:

Layer 1: signal

Layer 2: ground plane, unbroken

Layer 3: power plane/islands, signals

Layer 4: signal/power routing

Layer 5: ground plane, unbroken

Layer 6: signal

Dielectric thickness should be selected to support required signal trace characteristic impedances and power plane capacitance/inductance. Resonance analysis should be performed on all plane cavities.

1.2 Via padstack

Via padstack dimensions should support density requirements. Anti-pad diameter should be small enough to allow an adequate copper web between clearance holes of adjacent vias, and still meet PCB fabrication tolerances.

1.3 Part orientation and placement

The DRAM placement and orientation should be given priority over other unassociated components in order to minimize trace lengths. Closely following the signal trace routing guidelines allows trace lengths to be reduced.

1.4 Ground and power supply connections

For proper operation of devices, it is critical provide a very low impedance wide bandwidth connection to ground and the voltage supplies. To achieve this it is necessary to minimize inductance between the device power and ground balls and the PCB ground plane and decoupling network.

The above guideline also applies to other critical components: termination resistors, decoupling capacitors, ICs, multiple ground or power pins from the same IC.

Each ground ball should be directly connected to the PCB ground plane with its own via. Do not share vias between multiple ground balls. An exception to this rule is the center 6x6 ground ball grid, which should have a fully populated ground via grid between the balls and the surface layers can be filled.

Each power ball should be directly connected to the PCB decoupling network with its own via. Do not share vias between multiple power balls except when multiple power balls are adjacent to each other and are connected to the same voltage plane. In any case, use the maximum number of vias that space allows.

To avoid cross-contamination of ground or power supplies between different devices (for example, an IC and a termination resistor), multiple ground or power balls should not share ground connections. Each ball/pin should have their own via to the ground or power plane, not a shared via. Power and ground connections should not simply be connected to surface layer copper fill areas - these are not good low impedance paths at high-frequencies.

Ball to via trace: Each ground and power ball should be connected to it's via with a short, wide trace. It is critical to minimize trace length and maximize trace width. Trace length should be less than 1 mm in the ball field, 0.25 mm outside a ball field, Trace width should be wide. Do not simply connect ground or power balls to surface fill areas. A close, direct via to the ground or power plane is necessary. In cases where it is not possible to achieve this length, there is a relatively high impedance. Every effort should be made to minimize the length, and to consider only power connections requiring lower bandwidth for high impedance power connections.

Ferrite beads and RC filters: The perceived need for ferrite beads and RC filters is very often the result of poor grounding and power supply decoupling, resulting in excess noise on power rails. A well-designed power delivery network eliminates the need for these remedies.

2 DDR memory interface

2.1 DRAM power decoupling

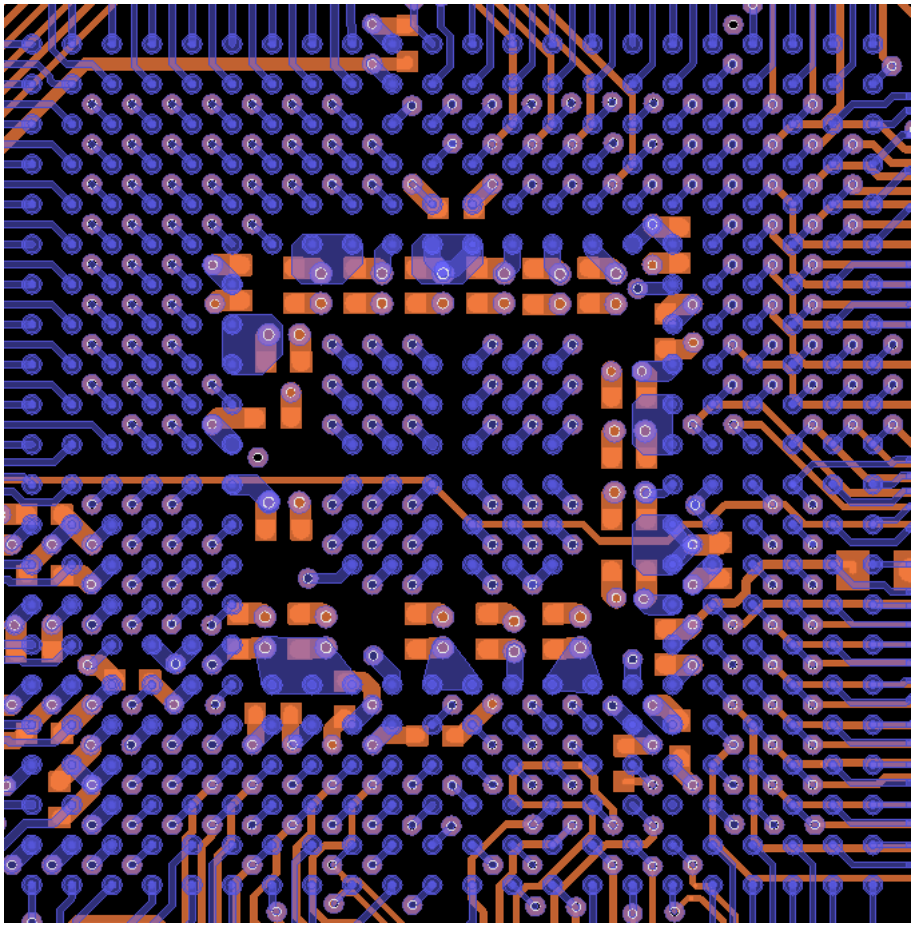
A low impedance wide bandwidth power delivery network (PDN) is critical for the proper operation of high-speed ICs such as SPEAr and DDR memory. If the PDN impedance is too high or does not have sufficient bandwidth, this affects the logic performance, resulting in ground and rail bounce, slower rise/fall times of both I/O and internal logic, which in turn result in delayed timing of events. These timing delays, from inadequate ground and power, subtract directly from the specified timing budget, which in turn can result in failure of the interface.

To achieve a low impedance, wide bandwidth power delivery network, it is critical to use appropriate decoupling capacitors and capacitor layout. A large portion of the power delivery network's frequency spectrum is above the series resonant frequency of the decoupling capacitors, where they are inductive. The PCB layout for decoupling capacitors is also inductive, and is a larger inductance than the capacitor itself. It is necessary to select capacitors with low inherent inductance (small package size), a lossy dielectric, and a PCB layout that provides the lowest possible inductance. For IC core voltage and high-speed I/O supplies (like DDR), as many capacitors should be used as can fit in the space available. This adds many parallel paths, reducing the overall inductance seen by the IC. A small capacitor package size and a small layout can be used to enable this.

Capacitors: Use 0402 package size to minimize the mounting inductance. The small 0402 package frees more board space, which is essential in high-density areas for more decoupling capacitors and signal routing. A capacitance value of 100 nF or larger is recommended, with X7R or X5R dielectric. Do not use Y5V dielectric for decoupling of mid-frequency applications. A few capacitors of smaller capacitance value probably may be necessary to suppress plane resonance. The correct value(s) to use for controlling resonance is very dependent on the board layout and stack up and must be determined individually for each unique PC board.

Decoupling capacitor layout: The layout of decoupling capacitors is extremely important to minimize the induction loop formed between the capacitor and the IC power and ground balls. The vias should be placed on the side of the capacitor lands, not the ends. The vias should be located at minimum keepout distance and connected to the capacitor lands with a wide trace - at least as wide as the via pad. Vias of opposite polarity should be placed as close together as possible (minimum keepout distance) and vias of the same polarity should be kept separated as much as possible. Following these layout guidelines can reduce the capacitor mounting induction loop by 50% or more over a layout with vias at the end of the capacitor lands. This is a very significant improvement, and is free. If space allows, a second pair of vias on the opposite side of the capacitor may be added to reduce the inductance further.

Figure 1. An example of an effective low-inductance decoupling capacitor location and mounting layout.



2.2 Data signal routing

Signal trace geometries:

Width: 4 mil to 5mil, 0.1 mm to 0.13 mm

Minimum space: 4 mil to 5 mil, 0.1 mm to 0.13 mm

Trace to trace spacing should be increased where routing density allows to reduce crosstalk. A minimum separation of 10 mils is recommended, except for BGA escape areas.

All signals should be routed referenced to a ground plane, not a power plane.

Do not route any DDR2 signals on top of split planes or voids of copper.

Do not route any other signals inside or on top of the area reserved for DDR.

Maintain adequate separation between DDR signals and any other signals.

Traces routed near the edge of a reference plane should maintain at least 30 mil gap to the edge of the reference plane.

To minimize the reflection all the traces should have an impedance of 55 Ohm ± 10%.

Data lanes:

DL0: Data Lane 0 includes DQ0-DQ7, LDQS-LDQSn, LDM.

Route these signals on the same layer(s), with similarly located layer transitions if they exist.

Length/timing skew: This set of signals traces must have their lengths matched to within 2.5 mm or 100 mils (approximately 15 ps).

DL1: Data Lane 1 includes DQ8-DQ15, UDQS-UDQSn, UDM.

Route these signals on the same layer(s), with similarly located layer transitions if they exist.

Length/timing skew: This set of signals traces must have their lengths matched to within 1.5 mm or 60 mils (approximately 10 ps).

Note: Trace lengths for different clock domains are not required to be matched. Trace lengths for DB0 do not need to match the trace lengths for DB1. Trace lengths for address/command signals do not need to match trace lengths for DB0 or DB1. Use minimum trace lengths within each clock domain.

CLK/CLK# and DQS/DQS#

CLK/CLK# and DQS/DQS# signals should be routed as differential pairs, length matched.

GATE_OPEN(0,1)

GATE_OPEN(0,1) must be left unconnected. Do not connect any trace, via, or test point to balls Y13 and Y17.

2.3 Trace length matching

The SPEAr600 package substrate does not have equal-length DDR signal traces. To insure timing alignment for all signals, both the substrate trace length plus the PCB trace length for each signal must be equalized to meet the trace length skew tolerance for all signals within the clock domain. A procedure for accomplishing this is provided below, along with a worksheet to facilitate this process during DRAM interface layout. The procedure may require an iteration if the initial longest trace must be lengthened to accommodate serpentine routing of other signals.

To determine trace length matching requirements for each signal, follow these steps, utilizing the worksheet provided. Ltol (maximum length difference within a clock domain) is 2.5 mm, or 100 mils.

1. Route signals per above guidelines, using the shortest possible length on all signals at this step (length equalizing is done later).
2. Tabulate the lengths of all traces that were routed in step 1, by filling in worksheet in the column labeled "INITIAL SHORTEST PCB TRACE LENGTH", and "PCB VIA LENGTHS".

Note: All signals within a clock domain should have an equal number of vias and equal via lengths, if so, then step 2 can be omitted.

3. For each clock domain, identify the trace with the maximum total length: max(trace length + delta_length + via length). Label this trace Lmax, it is the reference length that

other trace lengths must be adjusted to meet within Ltol. The worksheet computes Lmax for you.

4. With the longest total trace known (Lmax), the shortest total trace length permitted (Lmin) can be calculated by Lmin = Lmax-Ltol. Record the value of Lmin.

Note: Lmin is the total of PCB + substrate trace lengths. The worksheet computes Lmin for you.

5. Identify all signals within each data lane whose total length is less than Lmin. Tabulate this difference. The worksheet does this for you.
6. Add the length computed in step 5 to the traces. The worksheet does this for you.

Table 1. Package substrate trace length

Signal name	Ball	Package substrate trace length	
		(mm)	(mils)
DDR_ADD_0	AB3	9.0	354
DDR_ADD_1	AB4	8.2	323
DDR_ADD_2	AA4	7.5	295
DDR_ADD_3	Y4	7.3	287
DDR_ADD_4	W4	5.7	224
DDR_ADD_5	W5	5.7	224
DDR_ADD_6	Y5	6.4	252
DDR_ADD_7	AA5	5.9	232
DDR_ADD_8	AB5	7.6	299
DDR_ADD_9	AB6	8.3	327
DDR_ADD_10	AA6	5.6	220
DDR_ADD_11	Y6	5.9	232
DDR_ADD_12	W6	5.4	213
DDR_ADD_13	W7	4.6	181
DDR_ADD_14	Y7	5.4	213
DDR_BA_0	Y9	3.9	154
DDR_BA_1	W9	4.4	173
DDR_BA_2	W10	6.6	260
DDR_RAS	AB7	7.0	276
DDR_CAS	AA7	6.6	260
DDR_WE	AA8	5.7	224
DDR_CLKEN	AB8	7.0	276
DDR_CLK_P	AA9	5.4	213
DDR_CLK_N	AB9	5.8	228
DDR_CS_0	Y8	5.3	209
DDR_CS_1	W8	4.8	189

Table 1. Package substrate trace length (continued)

Signal name	Ball	Package substrate trace length	
		(mm)	(mils)
DDR_ODT_0	AB2	9.7	382
DDR_ODT_1	AB1	9.9	390
DDR_DATA_0	AB11	8.2	323
DDR_DATA_1	AA10	6.0	236
DDR_DATA_2	AB10	5.6	220
DDR_DATA_3	Y10	5.3	209
DDR_DATA_4	Y11	4.9	193
DDR_DATA_5	Y12	3.5	138
DDR_DATA_6	AB12	7.0	276
DDR_DATA_7	AA12	4.9	193
DDR_DQS_0	AB13	7.6	299
DDR_nDQS_0	AA13	6.6	260
DDR_DM_0	AA11	5.2	205
DDR_GATE_0	Y13	4.0	157
DDR_DATA_8	AB15	5.7	224
DDR_DATA_9	AA16	5.3	209
DDR_DATA_10	AB16	7.8	307
DDR_DATA_11	Y16	6.4	252
DDR_DATA_12	Y15	5.5	217
DDR_DATA_13	Y14	4.8	189
DDR_DATA_14	AB14	7.1	280
DDR_DATA_15	AA14	5.8	228
DDR_DQS_1	AB17	8.8	346
DDR_nDQS_1	AA17	7.6	299
DDR_DM_1	AA15	5.7	224
DDR_GATE_1	Y17	8.3	327

2.4 Trace length matching, dual DRAM configuration

Layout of dual DRAM configurations produces additional constraints on the layout to maintain adequate signal integrity and timing margin for a reliable interface. A "T" routing topology should be used, keeping the branch routes to the two DRAMs as short as possible. Length matching of traces to each device should be performed as described in *Section 2.3:*

Trace length matching. Additionally, matching the length of the branch traces to each DRAM is necessary. Mismatches in the branch length rapidly degrade signal integrity. The branch lengths, and therefore the spacing between two memory devices, should be minimized.

Figure 2 (A, B, C) below and *Table 2* describe the length matching guidelines for a dual DRAM topology.

Note: This is a guideline only, simulations should be performed using IBIS models on the actual PCB layout to assess signal integrity and timing margin.

Figure 2. Dual DRAM "T" routing topology

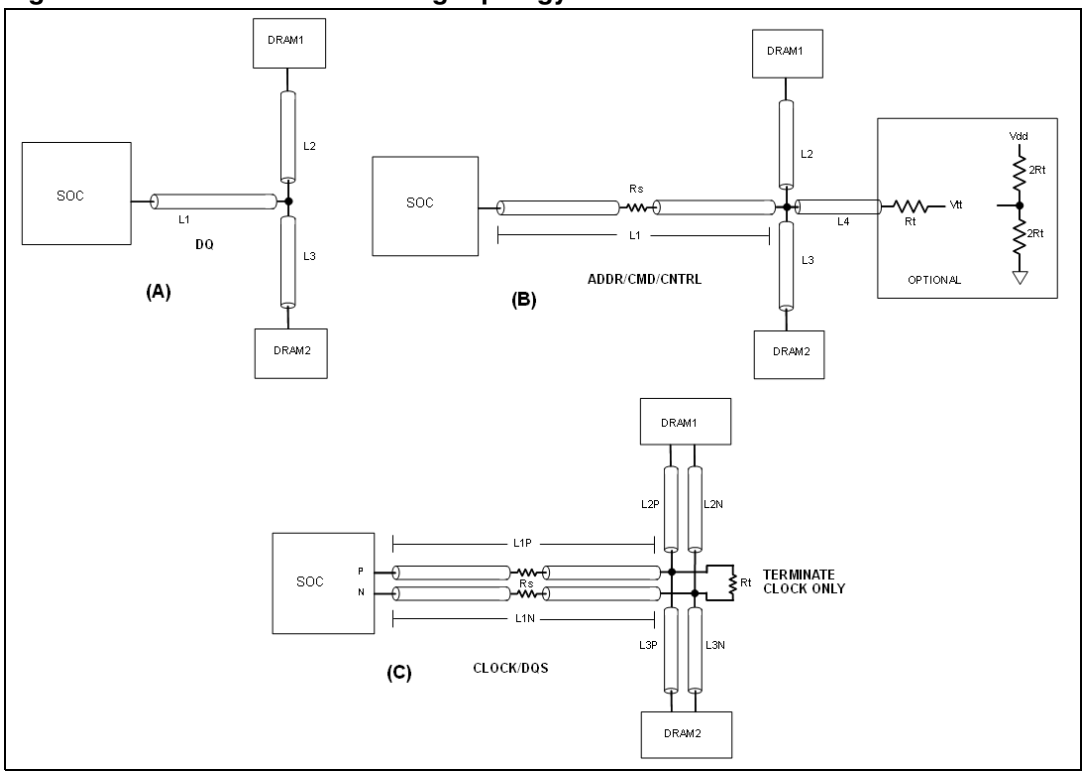


Figure 2 (A) shows the topology of the data signals, *Figure 2* (B) shows the topology of the address/command/control signals with optional shunt terminations, the *Figure 2* (C) shows the topology of the differential signal CLK and DQS. *Table 2* lists guidelines for length matching of the various segments.

Table 2. Trace length matching guidelines, dual DRAM configuration

Symbol	Description	Max	Units
tmm	L1 +L2, L1+L3 length matching for all signals	15	ps
Tmm2,3	|L2-L3| length matching tolerance of branches	30	ps

2.5 Return path integrity

A continuous path for return current must exist for all DRAM signals. If not, signal delays, significant crosstalk, and timing violations may likely result.

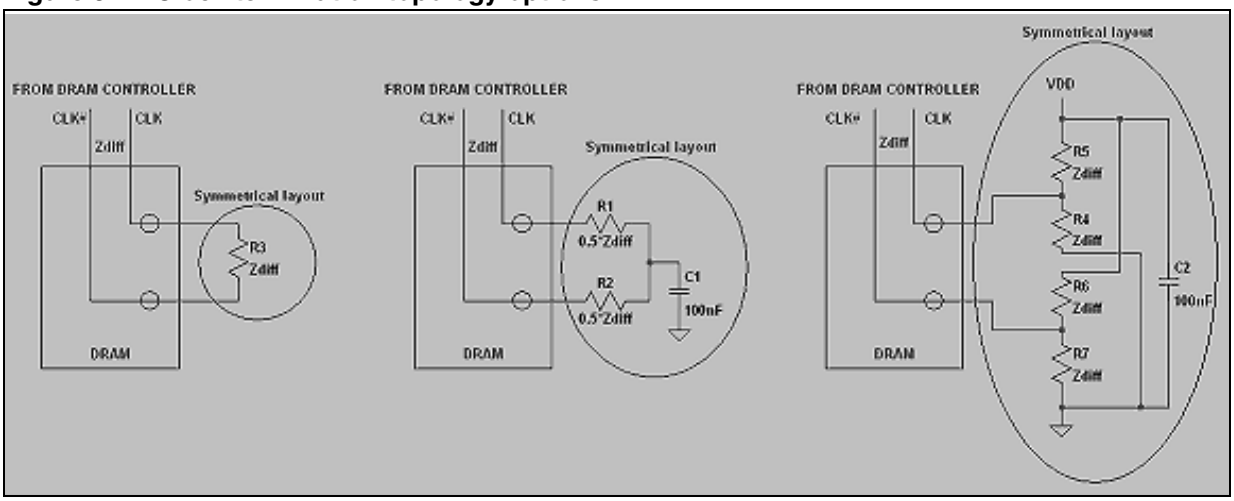
Signal layer changes: The preferred location for layer change vias are near the signal ball under the device, either DRAM or SPEAr, enabling a signal return path through the device ground vias and decoupling capacitors.

If layer changes through a via to a different reference plane are necessary away from the devices, the layer transitions should have a nearby path for return current. If both layers are ground, a ground via should be placed less than 1 mm from the signal via. Sharing return current vias should be avoided. Each signal via should have its own nearby return via (ground via). If multiple signals change layers in close proximity, each signal via should have its own return current via. So, signal vias (and their return current vias) should be separated from other signal vias with a stagger pattern. If routing density prevents a stagger pattern, add as many ground vias as possible among the signal vias.

2.6 Clock routing

The clock differential signals should be routed with a pair of matched length symmetrical traces with 110 Ohm differential impedance. The preferred clock routing layer is the top layer with no layer transitions. The termination layout should be symmetrical, without stubs, and located as close as possible to the clock balls on the DRAM. Three termination topologies are shown in *Figure 3*. Termination resistors and decoupling capacitors should be placed in a tight orientation to minimize trace lengths. Termination components should not share vias. The decoupling capacitor should have a via to ground placed against the side of its ground land. *Figure 3* shows three clock termination topology options. The middle topology provides the best tradeoff between part count and common mode rejection.

Figure 3. Clock termination topology options



2.7 Vref routing

An accurate and quiet Vref should be provided to both the DRAM and the controller. A noisy Vref effectively introduces jitter because of this it can be a significant source of jitter-caused timing errors. Vref is generated by a precision voltage divider. Recommended: use 0.1% tolerance resistors. A decoupling capacitor should be placed very close (within 1 mm) to the Vref balls. Use good capacitor layout technique. The voltage divider resistors should be placed close to the DRAM device to minimize trace length, but not so they interfere with other critical signal or power routing. The Vref trace should not be routed near noisy traces

or planes. Do not place a decoupling capacitor at the junction of the resistors - only at the Vref balls. If the Vref trace length must be long, the divider resistor value should be close to 2x the characteristic impedance of the Vref trace; 150 ohms should work well without consuming too much power. If a long trace and/or noise coupling results between the DRAM and controller, then it is preferable to generate a separate Vref for the DRAM and controller.

2.8 Observability

It is important to be able to observe important signals for system validation, timing, signal quality, and debugging. Place test points on any signals or set of signals that may be needed for these purposes. Always provide a ground via nearby any test points (<4mm) for the probe ground. Only very small test point pads can be used, preferably just a signal via. Or if there is insufficient space, simply a window in the solder mask over a trace can serve the purpose. Do not create test point structures that may significantly degrade signal quality, for example, large test points, stubs, etc.

Test points should be located at both ends of the trace (two test points per signal), as close to the device balls as practical (a via next to the ball is preferable, untented on the bottom). It is especially important to be able to observe signals at both the driving and receiving ends of a trace in high-speed interfaces to validate timing parameters, and quantify driver behavior and reflections. Observing a signal at only one end may hide important features that are evident at the other end, even with very short traces as is the case in a point-to-point DDR interface.

In a DDR memory interface the routing and component density is too high to add test points on all signals. A subset of DDR signals with test points is a good compromise. Test points for the following DDR signals should be included in all designs:

- CLK/nCLK
- DQS/nDQS (all data lanes)
- DQ (Select a small number of signals that are representative of best and worst signal paths, at least two DQ signals.)
- Address and Command (Select subset of signals that are of interest and that are representative of best and worst signal paths.)

3 USB interface

3.1 USB routing

USB signal trace routing should follow good high speed PCB rules and meet the specifications for differential impedance and maximum trace delay between connector and the SPEAr600 device.

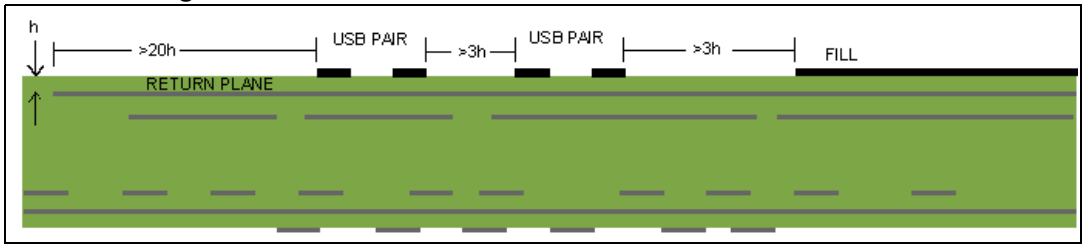
- USB data traces should be routed with the shortest most direct path possible to their connectors.
- USB data traces should have no resistors or magnetic components.
- USB data traces should be routed only over ground planes.
- USB data traces should never cross gaps or breaks in the return plane.
- USB data traces should never be routed under other devices or between the pins of other devices.
- USB data vias should be widely separated from other signal vias.
- If USB data traces transition layers to a different return plane, then ground vias for the return current should be placed very close to the signal vias.

Table 3. USB signal routing constraints

Symbol	Description	Min	Typ	Max	Units
ZODiff	Differential impedance(1)	81	90	99	Ω
t_{d_Dev}	Trace delay of device port(1)			1.0	ns
t_{d_Host}	Trace delay of host port(1)			3.0	ns
t_{d_match}	Trace length mismatch			0.15/3.8	inch/mm
	Number or length of stubs			0	
	Number of via transitions			1	
	Space to adjacent signal traces(2)	3			h(3)
	Space to adjacent area fill(4)	3			h(3)
	Space to edge of return plane	20			h(3)

1. Trace delay between SPEAr and USB connector (Universal Serial Bus Specification, Revision 2.0)
2. This includes other USB data trace pairs.
3. The unit 'h' is the thickness of the dielectric separating the trace from the nearest plane.
4. Do not use "guard" traces or ground flood/fill adjacent to high speed signal traces.

Figure 4. Cross section of PCB showing minimum spacing dimensions for USB signal traces



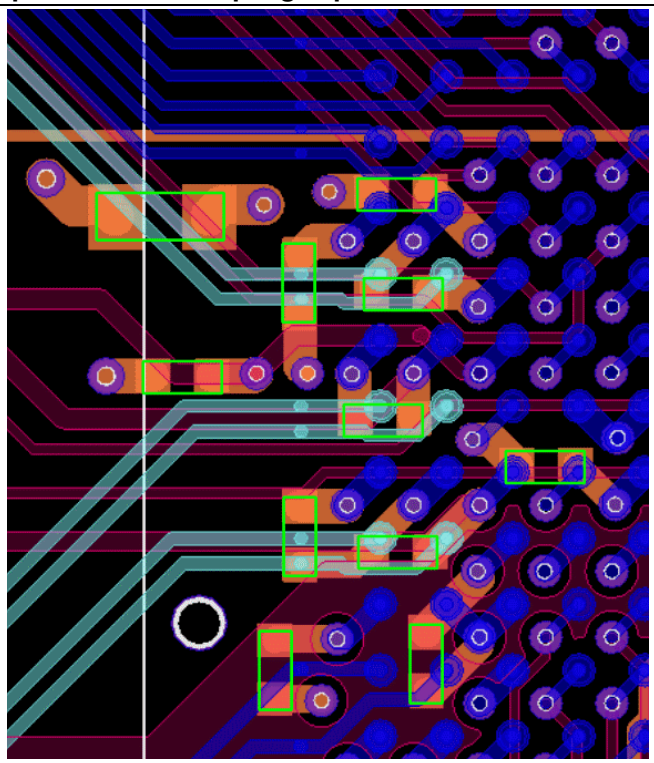
3.2 USB decoupling and reference resistor

Decoupling capacitors should be placed as close as possible to the power balls, preferably directly under the power balls using short wide connecting traces. Placing decoupling capacitors at a distance degrades the performance, which can result in interoperability problems and compliance violations with the specifications.

Table 4. USB power, ground, and reference guidelines

Pin	Guideline
V_{SS}	Connect all V_{SS} pins directly to the internal PCB ground plane
V_{DD}	Connect all V_{DD} pins to 100 nF capacitors under the ball using a short wide trace
USB_RREF	External reference resistor (1.5 kohm). Connect resistor to ground.

Figure 5. Example of USB decoupling capacitor bottom mounted layout



Legend:

Blue: top layer of PCB

Red: internal power/signal routing

Orange: bottom layer of PCB

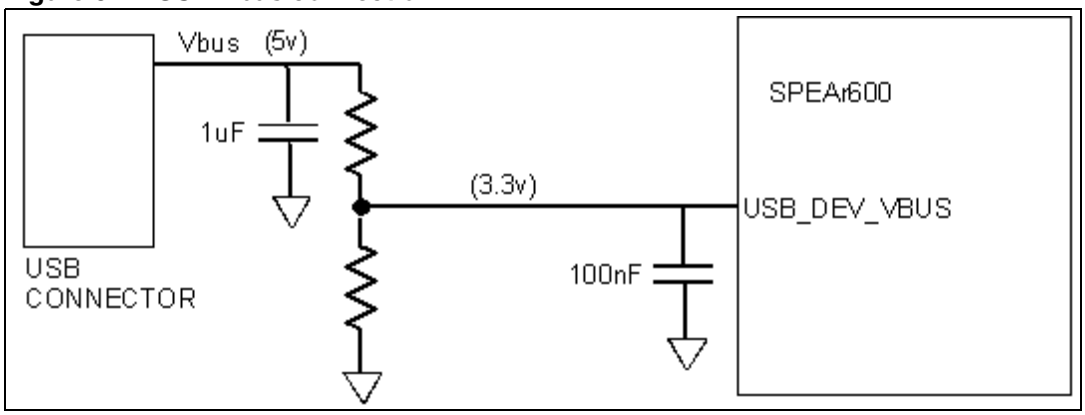
Cyan traces: USB data pairs.

Green rectangles: Position of the bottom mounted decoupling capacitors for the dedicated supplies.

3.3 USB Device Vbus

The 5 V USB Vbus signal on the device connector should be reduced with a resistor divider to 3.3 V to connect to the SPEAr600 USB_DEV_VBUS pin. A 100 nF decoupling capacitor should be placed near the SPEAr600 pin, and a 1 µF capacitor placed near the connector.

Figure 6. USB Vbus connection



4 GPIO and CLD interface considerations

There are no stringent requirements for GPIO or CLD interfaces, but some precautions should be considered for some applications that employ very wide buses switching simultaneously, or have electrically long interconnects, such as signal traces 1 ns or longer. These conditions apply heavy loading to the power distribution network supplying these I/Os, and it is imperative that good decoupling practices are used for the 3.3 V supply. See *Appendix A* for decoupling capacitor layout guidelines. When wide buses or long interfaces are necessary, 22 Ohm series resistors placed close to SPEAr provide additional benefit by reducing the loading seen by the I/Os.

5 Using an external clock

Using an external clock source to drive SPEAr600 is accomplished by using a compatible clock generator that meets the necessary jitter and waveform specifications, and configuring the internal clock for an external source.

The external clock signal is applied to oscillator MCLK_XI pin. This is a digital signal with a nominal voltage swing between 0 V and 2.5 V. If a clock source with a different voltage swing is used, then a voltage translator must be used. A 3.3 V signal should not be applied to the MCLK_XI pin.

5.0.1 External clock signal requirements

The clock signal specifications are listed in the table below.

Table 5. External clock signal requirements for oscillator MCLK_XI input

Symbol	Description	Min	Typ	Max	Units
V_{DD_OSC}	Oscillator power supply voltage		2.5		V
V_{IH}	High input voltage (1)	V_{DD_OSC} - 0.2			V
V_{IL}	Low input voltage (1)			0.2	V
t_{RISE}	Rise time of clock input signal	3		10	ns
t_{FALL}	Fall time of clock input signal	3		10	ns
f_{CLK}	Frequency of external source		30.000		MHz

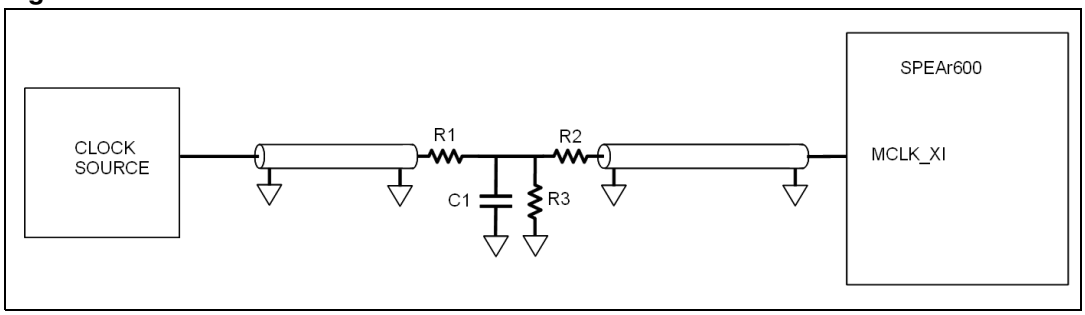
1. Because the MCLK_IN is the input to the oscillator gain stage, V_{IL} and V_{IH} levels and rise/fall time are more restricted than digital inputs.

5.0.2 Voltage translation of external clock

If the external clock's output has a signal swing of other than 2.5 V, then some form of level translation must be used on the signal before it is applied to MCLK_XI. An IC level translator can be used, or a resistor voltage divider network can be used if the source has a 3.3 V swing. The level translator must provide a stable, monotonic, and clean waveform to the clock input.

The resistor network must account for several conflicting requirements. First, the correct resistor ration must be used to provide the correct levels, and second, because the connection between the clock source and SPEAr600 is a trace on a PCB, which is a transmission line and therefore exhibits transmission line behavior, the resistor values used in the divider must adequately control reflections. The goal is to provide a clean waveform at MCLK_XI with minimal overshoot or ringing. *Figure 7* depicts the general network configuration.

Figure 7. External clock connection to SPEAr600



C1 is optional and may be needed for clock sources with extremely fast edges and is in the range of 10 pF to 33 pF. Some configurations may find that only R1 and R3 are needed (R2 = 0 Ohm).

The clock source IC and the resistor network should be placed as close to SPEAr600 as practical to improve signal quality. The resistor values depend on Ro (the output impedance of the clock source), the rise/fall time of the clock source, and the characteristic impedance and length of the transmission lines. Because every application has different variables, a single set of resistor values cannot apply to every situation. The resistor values are sensitive to Ro, which may not be known accurately from data sheets. Some experimentation may be necessary to arrive at suitable values and Spice simulations are recommended. *Table 6* lists values that can be used to start evaluation. These are examples of resistor network values with different output impedances of the clock source. Simulation and measurement of the actual circuit is strongly recommended.

Table 6. Examples of resistor network values

Ro	R1	R2	R3	Units
50	56	0	330	Ohms
50	56	56	330	Ohms
25	47	47	220	Ohms
15	56	56	220	Ohms

5.0.3 TDR test traces

Test traces should be add to all PCB designs on all signal layers. A single trace of nominal impedance between 8 to 15 cm long on each signal layer (it does not have to be straight) is simple to add to all designs and can always be placed where it does not impact the functional design, usually along the board perimeter. Include a test point pattern that matches your TDR probe. The test traces are invaluable to validate PCB impedance parameters.

5.0.4 Layer order check

A visual feature consisting of stair step numbered windows to verify layer order should be included in all PCB layouts. This is most commonly located along one edge of the board.

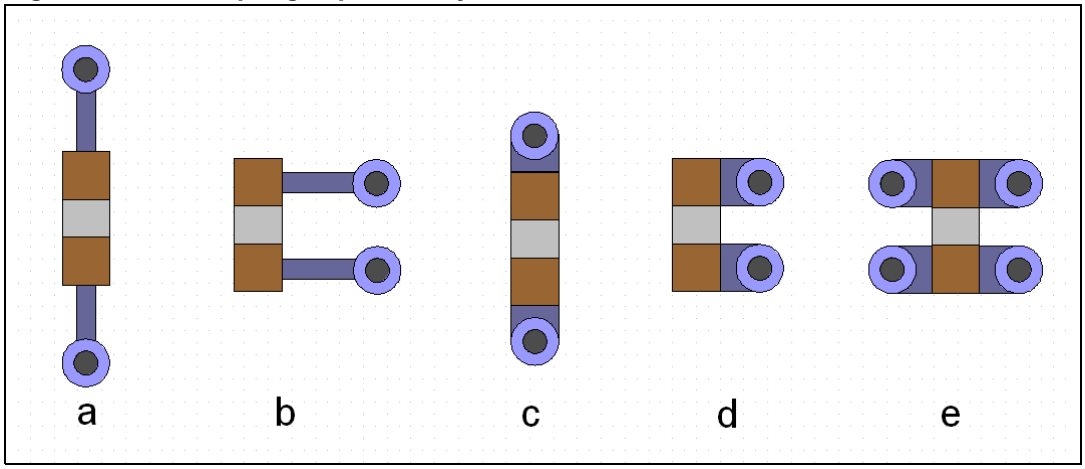
Appendix A Low-inductance decoupling capacitor layout

This appendix provides guidelines for low-inductance capacitor layout for high-frequency decoupling applications.

Figure 8 shows various layouts for decoupling capacitors. The layouts shown in (a) and (b) have inherently high inductance, and thus a high impedance at high frequencies and should be avoided.

The layouts in (d) and (e) have low inductance and should be used for high frequency decoupling applications. The 4-via layout (e) has the lowest inductance (but requires more board area), and should be used where space allows. The 2-via layout shown in (d) is a good compromise when space does not permit the 4-via layout. The layout shown in (c) should be used as a last resort when there is no space for (d) or (e).

Figure 8. Decoupling capacitor layouts



A.1 0402 compact land pattern

A area-efficient compact land pattern facilitates PCB layout of decoupling capacitors. *Figure 9* shows a commonly used land pattern.

Figure 9. Compact layout for 0402 size package

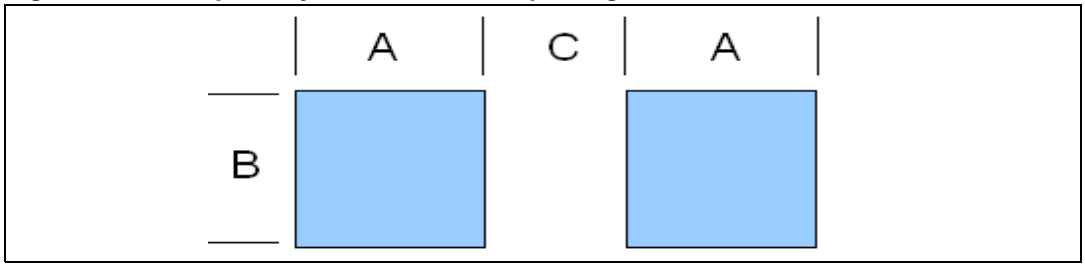


Table 7. Dimensions for compact 0402 land pattern

Dimension	Value	Units
A	20	mils
B	20	mils
C	15	mils

A.2 Low inductance layout for decoupling capacitors

Figure 10 shows a low inductance layout for a 0402 decoupling capacitor using 2 vias with a 10 mil drill size. The main features are

1. Vias placed close to the lands
2. Vias of opposite polarity place close together
3. Trace connecting land to via is wide.

Figure 10. Low-inductance layout for 0402 package.

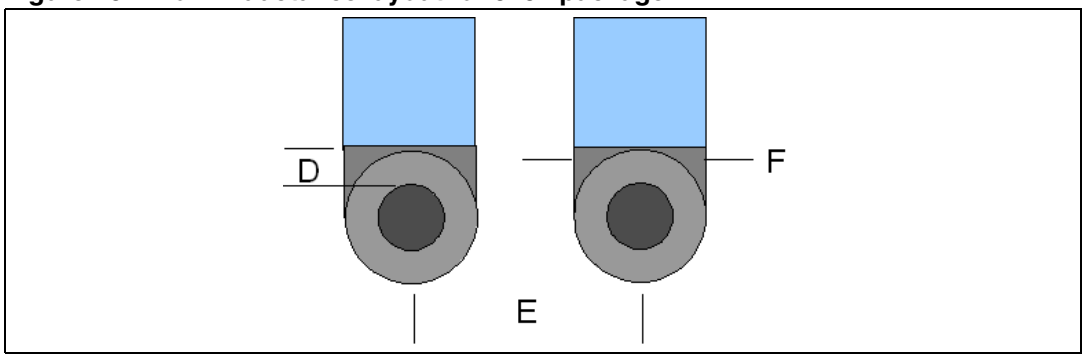


Table 8. Dimensions for a low inductance capacitor layout

Dimension	Value	Units
D (land to hole)	Fabrication tolerance dependent(1)	mils
E (hole to hole)	Fabrication tolerance dependent(2)	mils
F (trace width)	20	mils

1. The land to hole separation is determined by the PCB fabrication tolerances. This is usually 8 - 10 mils.
2. This distance should be minimized, consistent with PCB fabrication tolerances, typically about 30 mils. If the capacitor is placed within a BGA ball field, dimension E should be the same as the ball pitch

Revision history

Table 9. Document revision history

Date	Revision	Changes
02-Dec-2009	1	Initial release.

Please Read Carefully:

Information in this document is provided solely in connection with ST products. STMicroelectronics NV and its subsidiaries (“ST”) reserve the right to make changes, corrections, modifications or improvements, to this document, and the products and services described herein at any time, without notice.

All ST products are sold pursuant to ST’s terms and conditions of sale.

Purchasers are solely responsible for the choice, selection and use of the ST products and services described herein, and ST assumes no liability whatsoever relating to the choice, selection or use of the ST products and services described herein.

No license, express or implied, by estoppel or otherwise, to any intellectual property rights is granted under this document. If any part of this document refers to any third party products or services it shall not be deemed a license grant by ST for the use of such third party products or services, or any intellectual property contained therein or considered as a warranty covering the use in any manner whatsoever of such third party products or services or any intellectual property contained therein.

UNLESS OTHERWISE SET FORTH IN ST’S TERMS AND CONDITIONS OF SALE ST DISCLAIMS ANY EXPRESS OR IMPLIED WARRANTY WITH RESPECT TO THE USE AND/OR SALE OF ST PRODUCTS INCLUDING WITHOUT LIMITATION IMPLIED WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE (AND THEIR EQUIVALENTS UNDER THE LAWS OF ANY JURISDICTION), OR INFRINGEMENT OF ANY PATENT, COPYRIGHT OR OTHER INTELLECTUAL PROPERTY RIGHT.

UNLESS EXPRESSLY APPROVED IN WRITING BY AN AUTHORIZED ST REPRESENTATIVE, ST PRODUCTS ARE NOT RECOMMENDED, AUTHORIZED OR WARRANTED FOR USE IN MILITARY, AIR CRAFT, SPACE, LIFE SAVING, OR LIFE SUSTAINING APPLICATIONS, NOR IN PRODUCTS OR SYSTEMS WHERE FAILURE OR MALFUNCTION MAY RESULT IN PERSONAL INJURY, DEATH, OR SEVERE PROPERTY OR ENVIRONMENTAL DAMAGE. ST PRODUCTS WHICH ARE NOT SPECIFIED AS "AUTOMOTIVE GRADE" MAY ONLY BE USED IN AUTOMOTIVE APPLICATIONS AT USER’S OWN RISK.

Resale of ST products with provisions different from the statements and/or technical features set forth in this document shall immediately void any warranty granted by ST for the ST product or service described herein and shall not create or extend in any manner whatsoever, any liability of ST.

ST and the ST logo are trademarks or registered trademarks of ST in various countries.

Information in this document supersedes and replaces all information previously supplied.

The ST logo is a registered trademark of STMicroelectronics. All other names are the property of their respective owners.

© 2009 STMicroelectronics - All rights reserved

STMicroelectronics group of companies

Australia - Belgium - Brazil - Canada - China - Czech Republic - Finland - France - Germany - Hong Kong - India - Israel - Italy - Japan - Malaysia - Malta - Morocco - Philippines - Singapore - Spain - Sweden - Switzerland - United Kingdom - United States of America

www.st.com